Design and Optimization of a MEMS Monopropellant Micro-Thruster

Jeff Kujawa[1], Chuck Zakrzwski[2], Darren Hitt[3]
[1] The 2003 NASA Academy, Goddard Space Flight Center, Greenbelt, MD 20771
[2]Propulsion Branch, code 597, NASA Goddard Space Flight Center, Greenbelt, MD 20771
[3]Microfluid Mechanics Laboratory, The University of Vermont, Burlington, VT 05401

Completed as part of the 2003 NASA Academy midway report
July 17, 2003

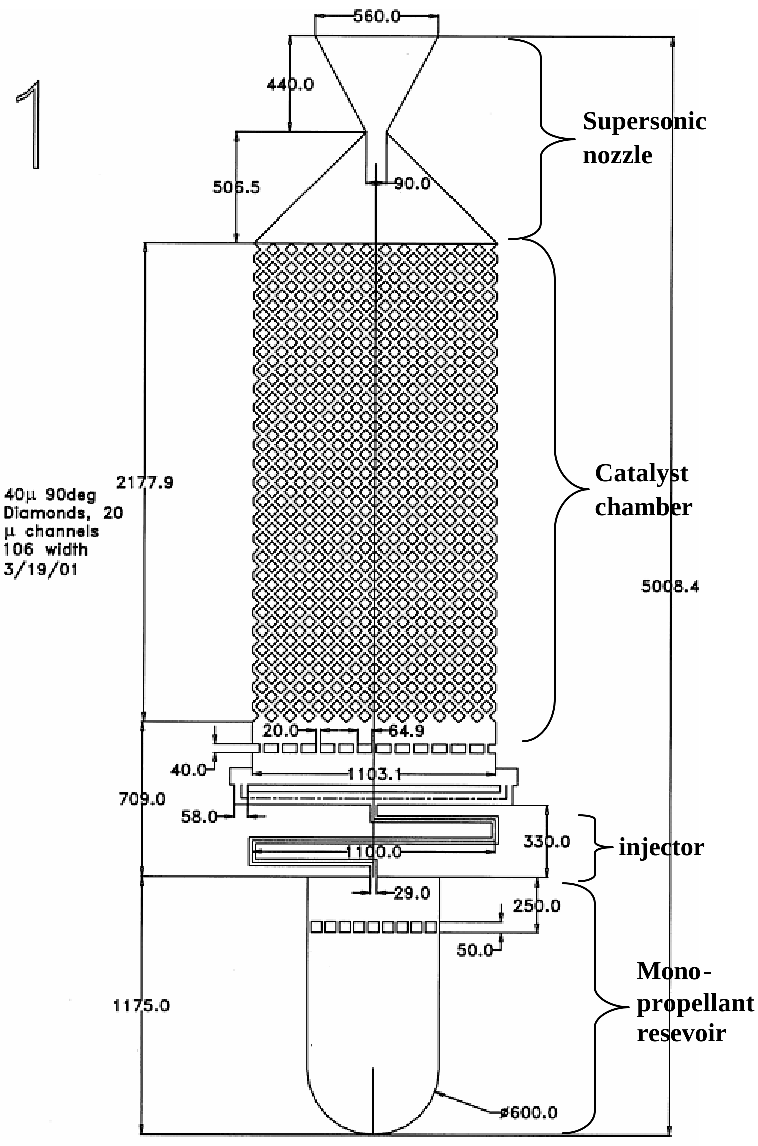

Figure 1. A schematic of a prototype MEMS based monopropellant micro-thruster. The geometry is 300 μm deep, formed by Deep Reactive Ion Etching (DRIE). A glass cover is then anodically bonded to seal the thruster.

Introduction

"Nanosats," or satellites featuring a mass of approximately 1-10 kg, are being considered for a wide range of advanced missions such as precision formation flying. Unique thrust requirements accompanying the nanosats have initiated a design concept for a MEMS-based microthruster. Due to the complexity of the fluid dynamics inherent on the micrometer scale, experimental and numerical studies have been proposed to better understand the flow properties as well as optimize the design.

The most recent prototype (refer to Figure 1) contains four main components. Starting from a monopropellant reservoir, high test hydrogen peroxide (HTP) is released via a micro-valve (not shown) into the thruster inlet port. From the inlet port, the propellant passes through an injector, which reduces the pressure to alleviate backflow issues. The HTP undergoes chemical decomposition in the catalyst chamber releasing a large amount of heat. The gaseous products are then accelerated to supersonic conditions in the converging/diverging nozzle.

Complex fluid dynamics inherent on the micrometer scale initiates a computational study of the supersonic nozzle as well as an experimental study of the catalyst bed. The hope is to optimize the geometry and achieve complete chemical decomposition.

Computational Analysis of the Supersonic Nozzle

Introduction

The nozzle is a key component in the overall performance of the thruster. Due to the high surface area to volume ratio common to the small geometry, viscous and thermal boundary layers may propagate well into the flow. As a result, the thruster's efficiency can be severely degraded. Computational fluid dynamics is used to analyze the operation and then optimize the nozzle design. To evaluate the effects of heat transfer, the interior nozzle walls are defined with a constant temperature based on the relatively high thermal conductivity of silicon (refer to Figure 2). A comparison between adiabatic conditions versus isothermal walls is reported, as well as suggestions for the divergence angle optimization.

Areas of Interest Include

- Viscous effects, boundary layer growth
- Impact on the thrust vector
- Optimization of the divergence angle
- Impact of heat transfer through the nozzle
    - Adiabatic vs. isothermal walls

Computational Methodology

- FLUENT 6® was used for all calculations
- Continuum modeling using a mixture of the product gases
- Constant mass flow rate and adiabatic flame temperature defined at inlet, pressure outlet set as far field
- Convergence was determined via residuals, monitors, and mass conservation

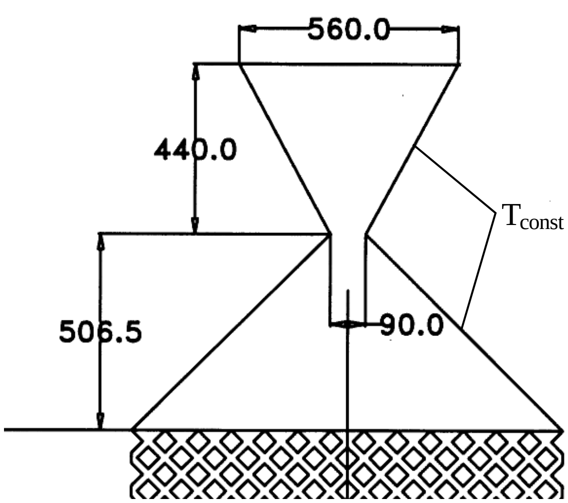

Figure 2. A close-up of the nozzle section showing the boundary conditions used for the simulations.

In order to study the effects of heat transfer, the nozzle walls are defined with a constant temperature (see Figure 2), which are then compared to adiabatic conditions. Reported is a wall temperature held at 200 K, for a pressure outlet of 1 kPa. The temperature and pressure where chosen as first steps towards approaching the actual operating conditions in space. Further reducing the magnitude of both boundary conditions is currently underway, with the hope of establishing a trend to ensure an accurate solution.

Due to the high surface area to volume ratio, a 2-dimensional model cannot be used to accurately depict the flow characteristics, especially the heat transfer effects[1]. However this model requires much less computation time and therefore is used as a baseline for the 3-dimensional model.

2-Dimensional Results Including Heat Transfer

The ultimate goal is to determine the effects of heat transfer on the thrust. The exit velocity is of particular interest in determining the heat transfer effects due to its relationship to the thrust a shown in equation (1).

$$F \approx \dot{m}\overline{V}_{exit} \qquad (1)$$

Figure 3 shows the difference in the exit velocity near the walls.

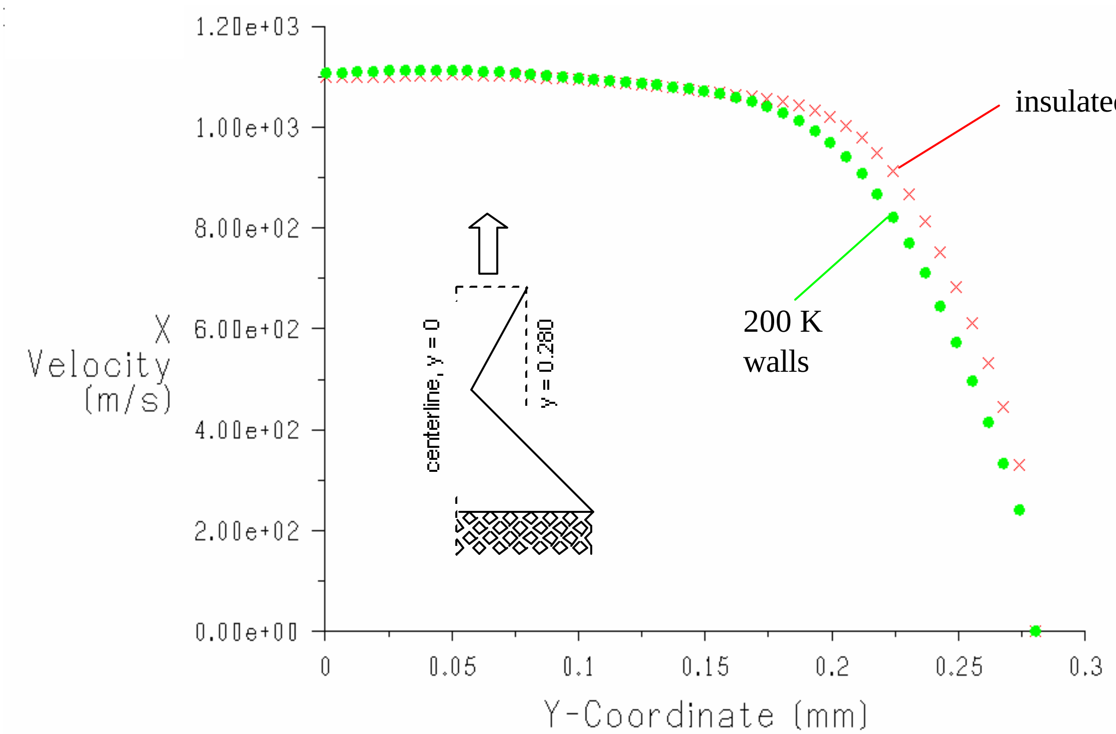

Figure 3. A plot of x-velocity versus the position in the y-direction at the nozzle exit for ½ the nozzle geometry (y = 0 marks the centerline). Notice the discrepancy in the velocity near the wall.

The effects of the constant temperature boundary condition are apparent, however do not seem to propagate far into the flow (see Figure 3-4). An overall thrust reduction of approximately 3% was observed.

Figure 4 supports the belief of a thermal region confined to the wall and Figure 5 shows the velocity contours. Notice the expansion at the exit plane, showing an underexpanded exit gas. If in fact the flow can be further expanded, increasing the divergence angle of the nozzle would increase the efficiency.

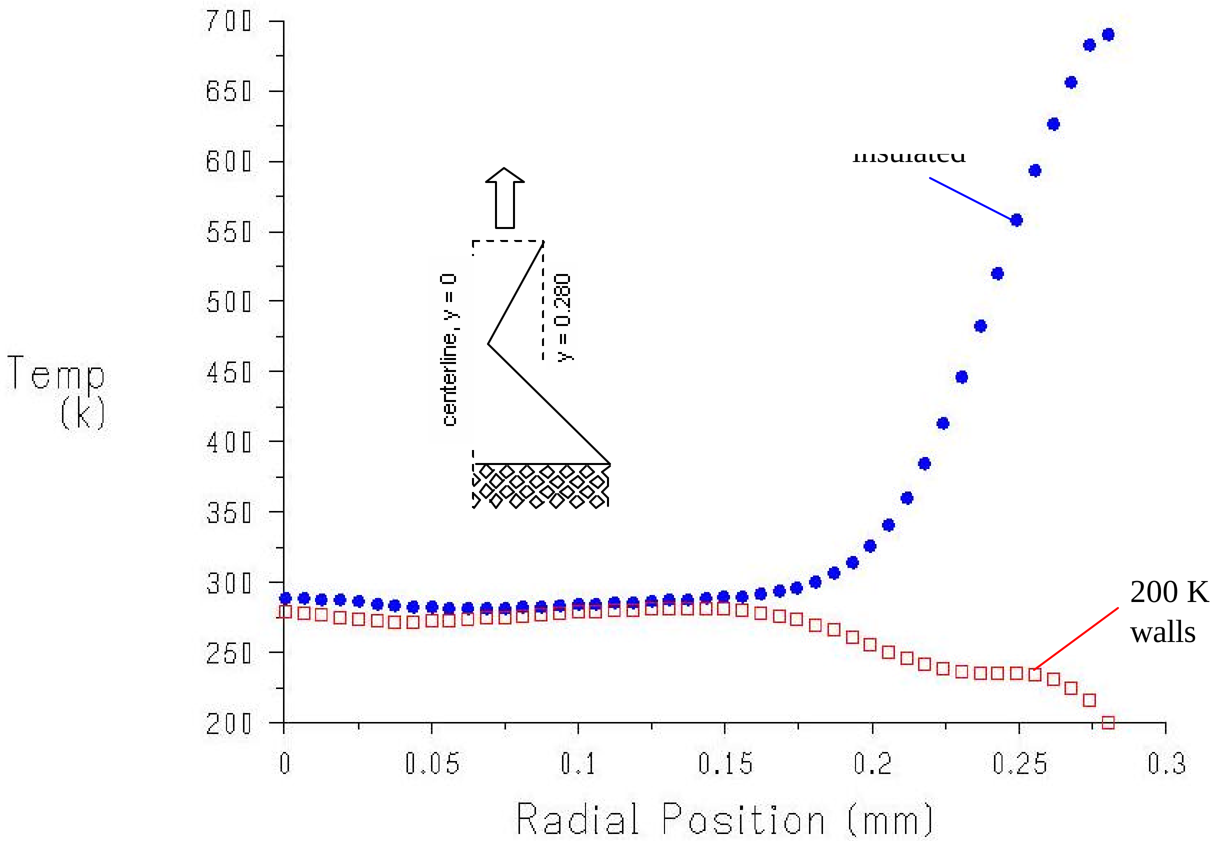

Figure 4. A temperature plot at the exit of ½ the geometry showing the thermal effects near the walls.

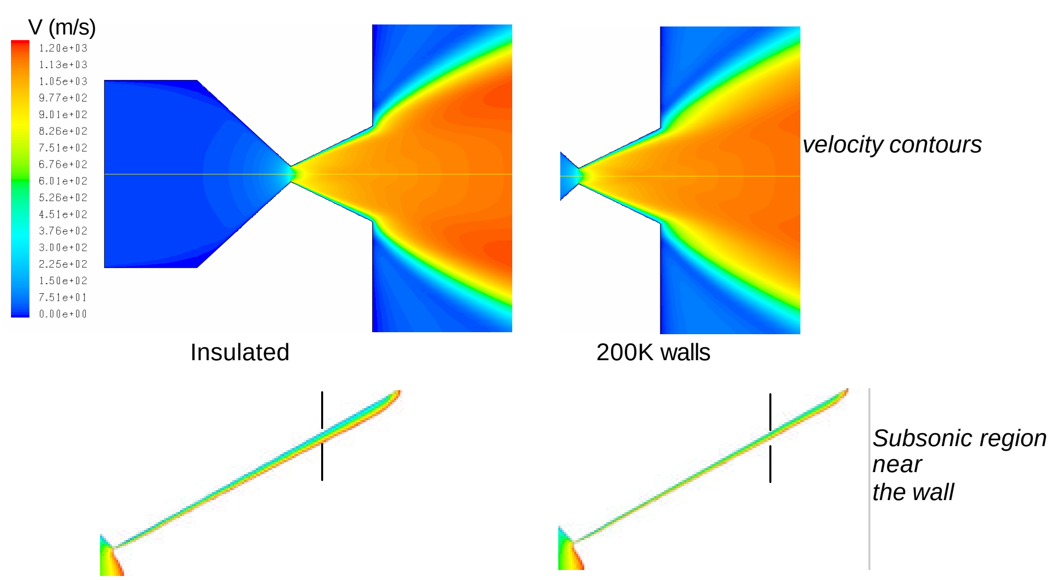

Figure 5. Velocity contours of the 2-dimensional nozzle. The subsonic region near the wall seems to be intimately linked to the efficiency of the nozzle.

3-Dimensional Results Including Heat Transfer

Consistent with the 2-dimensional results, heat transfer reduces the thruster's efficiency. 10% of the overall thrust is lost when applying 200 K walls.

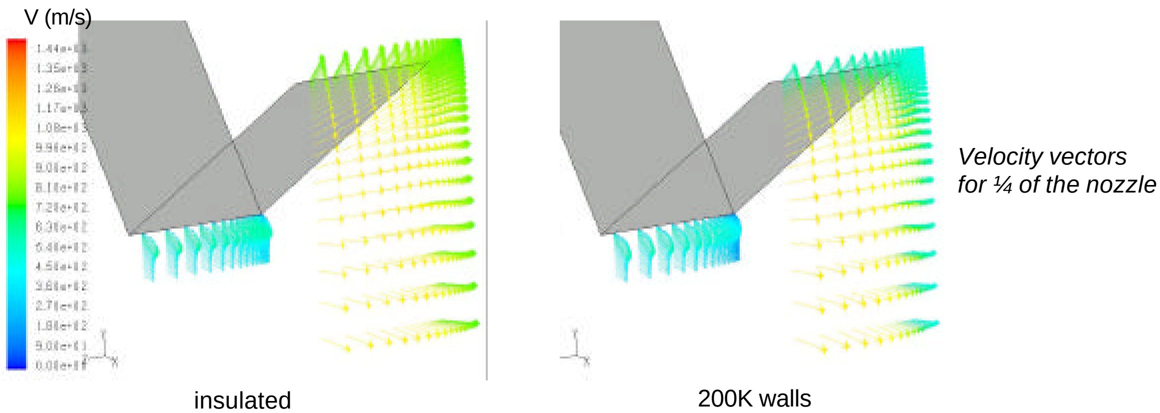

Figure 6. Velocity vectors showing the effects of heat transfer near the walls. Note, only ¼ of the geometry is shown.

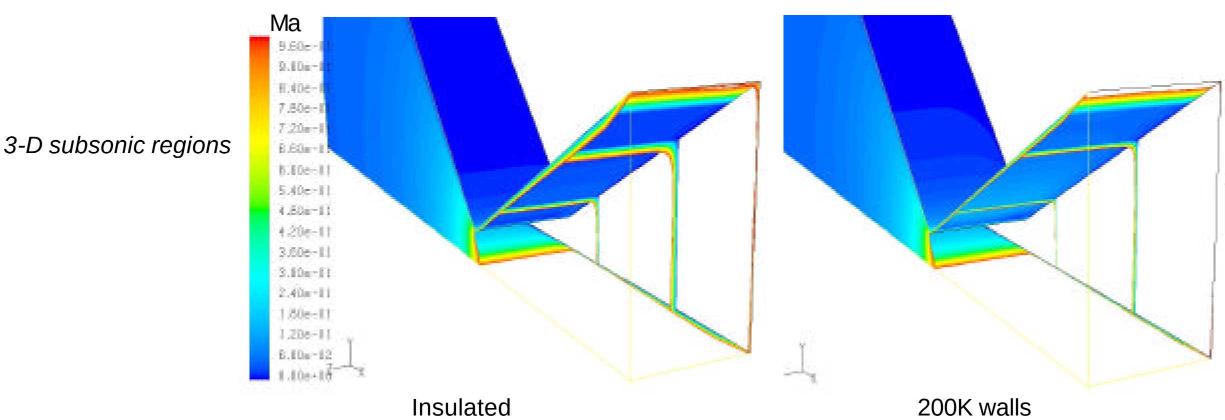

Figure 7. Subsonic regions near the wall for ¼ of the geometry.

Experimental Study of the Catalyst Bed

Introduction

Incomplete decomposition of the monopropellant is severely hindering the thruster's performance. The resulting 2-phase flow gives rise to an empirical study of the catalyst bed to determine the required decomposition length as well as the feasibility of using silver powder as the catalyst.

Experimental Setup

The setup being constructed is designed to pass HTP through a silver powder test bed. The catalyst is located inside a glass capillary tube fitted on one end with a 24 µm steel mesh/filter to prevent the silver from being lost.

- Capillary catalyst bed – Kimax® capillary tube (I.D. ~ 1.24 mm)
- Silver powder used as catalyst
- Hamilton® precision syringe and precision syringe pump produce a pre-defined mass flow rate

Test setup

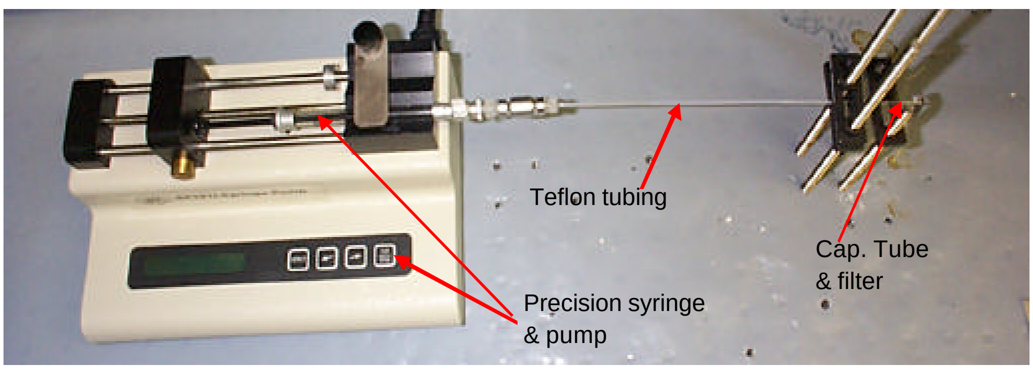

Figure 8. Proposed experimental setup. Teflon tubing was used due to its compatibility with hydrogen peroxide. The outlet gases are to be collected via condensation and analyzed with the Brix refractometer.

Experimental Methodology

The decomposition length is determined based on the exhausting products and the % of remaining HTP. Various catalyst bed lengths, material compositions, initial temperatures, and various flow rates will all be tested. The heat produced from the reaction is expected to vaporize the H_2O to produce fully gaseous products.

$$H_2O_2(l) \; ? \quad 2H_2O(l) + O_2(g) + \text{heat}$$

Methods of measurement include:

- % HTP measured using a Brix refractometer
- visual inspection for 2-phase flow

Conclusions

To date, 2-D and 3-D numerical models have been successfully completed and are in good agreement.

- The flow in the diverging region of the nozzle is underexpanded, suggesting a larger divergence angle.
- The 3-D case shows approximately a 10% reduction in thrust when considering heat transfer.
- The subsonic region found near the wall seems to be intimately linked to the efficiency of the nozzle.

Experimental tests are currently underway to characterize the catalyst bed length for a silver powder catalyst.

Future Work

Both numerical and experimental studies are proposed for the near future.

- Complete a grid independent study of the numerical model (currently underway)
- Further evaluate non-continuum effects to ensure a continuum based algorithm will accurately capture the flow properties. If non-continuum effects are found non-negligible, correct via source terms such as the Burnett Equations.
- Complete a conjugate heat transfer model to observe the effect of the silicon substrate.

- Evaluate the reaction dependence on temperature, packing density, and powder size.
- Complete a similar study of Alumina/Platinum powder mixture to determine its feasibility as a HTP catalyst.

References

[1]Alexeenko A, Levin D 2002 Numerical Modeling of Axisymetric and Three-Dimensional Flows in Microelectromechanical Systems Nozzles *AIAA* **40** 897-904

Acknowledgements

The cooperation of the staff of the NASA Goddard Space Flight Center Propulsion Branch, Code 597 is greatly appreciated.

The Vermont Space Grant Consortium for supporting this program.